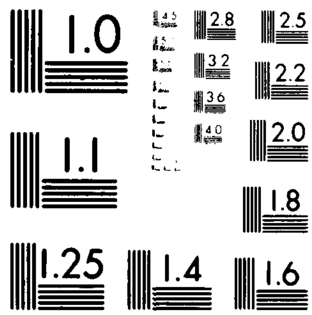
AD-A114 807 AERONAUTICAL RESEARCH LABS MELBOURNE (AUSTRALIA) F/G 12/1
A REVIEW OF COMPUTATIONAL FLUID DYNAMICS AND ITS IMPACT ON A.R.--ETC(U)
MAR 82 B D FAIRLIE
UNCLASSIFIED ARL/AERO-TM-336 NL

END
DATE
FILMED
6 82
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

UNCLASSIFIED

(12)

ARL/AERO-TECH-MEMO-336 AR-002-342

AD A114807

DEPARTMENT OF DEFENCE

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION

AERONAUTICAL RESEARCH LABORATORIES

MELBOURNE, VICTORIA

Aerodynamics Technical Memorandum 336

A REVIEW OF COMPUTATIONAL FLUID DYNAMICS AND ITS IMPACT ON A.R.L.

B.D. FAIRLIE

Approved for public release

COPY No

82 05 25 021

MARCH, 1982

UNCLASSIFIED

DEPARTMENT OF DEFENCE

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION

AERONAUTICAL RESEARCH LABORATORIES

Aerodynamics Technical Memorandum 336

A REVIEW OF COMPUTATIONAL FLUID DYNAMICS AND ITS IMPACT ON A.R.L.

B.D. FAIRLIE

SUMMARY

This memorandum reviews the major developments in computational fluid dynamics. The present and future capabilities of numerical simulations are assessed to determine the demands that they will place on available computer power. The impact that these developments will have on the use of computational fluid dynamics at ARL is considered together with their implications for the provision of new computers at ARL.

POSTAL ADDRESS: Chief Superintendent, Aeronautical Research Laboratories,
P.O. Box 4331, Melbourne, Victoria, 3001, Australia.

CONTENTS

INTRODUCTION

In August 1980 the Government agreed to Recommendation 52 of the Australian Science and Technology Council (ASTEC) Report "That the Department of Defence be asked to develop a detailed long-term plan for upgrading and extending facilities for R&D in aeronautics and aerospace, consulting with other government departments and agencies, including those with responsibilities in civil aviation, and with industry in the preparation of the plan" (Paragraph 11.2.4 Vol. 1B, Science and Technology in Australia, 1977-8). As part of that overall plan, consideration is being given to national needs for wind-tunnel facilities and how future requirements can best be satisfied, either by upgrading existing tunnels, which are now 30 to 40 years old, or by providing new tunnels.

Since upgrading or providing new, more capable wind tunnels would involve major expenditure, it is appropriate to consider whether the tremendous developments in the field of numerical methods for flow computation over the past two decades can, or will, obviate the need, either in part or in full, for this expenditure. Some of the more optimistic proponents of numerical flow computation have already suggested the wind tunnel will become obsolete in the not too distant future. The developments in numerical methods have resulted directly from the explosive growth in the speed and storage capacity of modern computers, and are of sufficient magnitude to constitute what might be called a new scientific discipline - computational fluid dynamics.

In addition to the study of wind tunnel needs, consideration is also being given to a major upgrading of the central computer (DECSystem10) at ARL and the speed and memory capacity of the computer should be determined, at least in part, by developments in computational fluid dynamics and the extent to which ARL will be involved in this field in the future. Both the computing load and the size and complexity of problems being solved on the present computer have increased rapidly during the past decade, and even though the system was expanded in 1975, it is now overloaded and has inadequate memory capacity to satisfy present requirements. In the past, decisions on speed and memory capacity of new computers have been based primarily on extrapolation of historical usage. This approach, although previously successful, appears inadequate for present decisions at a time of extremely rapid advances in computational methods and especially computational fluid dynamics.

Some feeling for the dramatic increase in the power of computers in the past twenty years can be obtained from an example quoted by Chapman[1]. A numerical simulation of the flow over an aerofoil using the Reynolds Averaged Navier-Stokes equations can be conducted on today's supercomputers in less than half an hour at a cost of around $1000. If such a simulation had been attempted twenty years ago on the best of the computers available at that time, the estimated cost would have been roughly $10 million, and the results would not be available for a further 10 years from now, since the computation time would have been about 30 years!

Increase in computer power have been accompanied by equally remarkable improvements in the efficiency of numerical algorithms for use on a particular computer. These two trends have compounded to give an extraordinary increase in the cost effectiveness of computational fluid-dynamics. This has been most evident in the increased complexity not only of geometry of the problems which have been feasible to compute, but also in the accuracy of representation of the complete equations of motion. Most of the early numerical solutions were based firmly on linear approximations to the equations governing inviscid fluid flows. At the present time non-linear (i.e. transonic) inviscid solutions can be routinely computed over fairly simple geometries, and there is a realistic possibility of computing non-linear viscous flows over complex three-dimensional aircraft geometries in the not too distant future.

The objectives of this paper are to review some of the major developments in and capabilities of computational fluid dynamics, and to consider the implications that developments in this field have on the need for wind tunnels and the speed and capacity of new computers for DSTO establishments.

THE ROLE OF COMPUTATIONAL FLUID DYNAMICS

The aims of computational fluid dynamics can be summarised as follows:-

1. To provide flow simulations which are either impossible or impractical to obtain in ground based facilities, for example the simulation of full scale flight Reynolds numbers or flows with significant chemical reaction.

2. To reduce the time and cost required to obtain the aerodynamic data necessary for flight vehicle design, development, modification or assessment.

3. Eventually to provide more accurate simulations than can be obtained in wind tunnels due to their inherent limitations on accuracy, caused for example by wall and sting interference or non-representative aeroelastic distortions.

What then are the relative roles played by the wind tunnel and computational fluid dynamics in the present and future solution of flow simulation problems? Rather than the imminent demise of the wind tunnel predicted by some, it would appear that both wind tunnels and computational methods will continue to play complementary roles, at least for the forseeable future. Figure 1[2] compares the attributes of wind tunnels with those of numerical methods. The wind tunnel deals with the complete, if not the desired physics, and allows study of most if not all the phenomena that may occur in the flow about a flight

vehicle. The computer can presently only give solutions to an approximation to the equations of motion, and although these approximations will improve with increases in computer capacity, it is doubtful whether the exact equations will ever be amenable to computation. The computer can however provide solutions in which the geometry and boundary conditions are accurately represented, unlike the situation in the wind tunnel where the existence of wind tunnel walls, model supports and flow nonuniformities produce flow fields which are not truly representative of free flight. Wind tunnel models are time consuming and increasingly expensive to manufacture, severely limiting the range of configurations which may be tested economically. However, once a wind tunnel model exists it is a relatively simple matter to change the flow conditions, the cost per data point actually reducing as more data points are accumulated. In the computer on the other hand, model geometry can be relatively easily changed, but changes in flow conditions may not be straightforward if a change in computational model is required (e.g. as angle of attack is increased a new model which correctly treats flow separations may have to be included). Also the cost of computer simulations tend to be dominated by the cost of actual computer time and will not reduce greatly as more cases are run. The ease of changing model geometry in the computer also allows fairly straightforward changes in boundary conditions e.g. the presence of stores or an external interfering flow field. Such changes are not easily included in wind tunnel simulations unless the model was originally designed with such changes in mind.

From an aircraft designers point of view, probably the most important and also unique aspect of the computational approach is the ability to consider not only the "direct" problem (i.e. to determine the flow around a given geometry) but also the "inverse" or "design" problem of determining the shape which produces a given pressure distribution. Recent developments combine such "inverse" solutions with some form of optimisation, and it is probably in this area that the (potential) benefits of computational fluid dynamics will best be exploited in the future.

DEVELOPMENT OF COMPUTATIONAL FLUID DYNAMICS

The development of computational fluid dynamics in the early part of this century was slow, and limited to the tedious hand calculation of linear inviscid approximations to the full equations of motion. Geometries were necessarily confined to simple usually two-dimensional, configurations. With the increasing availability of the electronic computer in the 1950's the range and accuracy of computational solutions increased enormously, and this period really marks the beginning of computational fluid dynamics as it is known today.

For computational methods to be usefully employed in simulating a physical situation two conditions must be satisfied:-

(i) The physics of the situation must be well enough known to be represented by an accurate mathematical model, and

(ii) Computers of sufficient capacity and speed must be available to solve the mathematical model in a practical time and at reasonable cost.

The situation for computational fluid dynamics in regard to these two conditions was summarised by Bradshaw[3] in 1972:

"In turbulence studies we are fortunate in having a complete set of equations, the Navier-Stokes equations, whose ability to describe the motion of air at temperatures and pressures near atmospheric is not seriously in doubt (it is easy to show that the smallest significant eddies are many times larger than a molecular mean free path). We are unfortunate because numerical solution of the full time-dependent equations for turbulent flow is not practicable with present computers".

Because the Navier Stokes equations are so highly nonlinear, this statement remains true today despite the rapid advances in computer technology since 1972, and their exact solution will remain virtually impossible. However, as computing power increases with time, better and better approximations have and will be included in their numerical representation.

The following sections will examine the major states of successively refined approximation to the full Navier Stokes equations from the simple linearized inviscid approach of early approximations to the large eddy simulations of future models. Each stage will be examined from the points of view of approximations used, applicability to real flows, availability of usable codes and demands on computer power to produce economic solutions. To aid in the understanding of the mathematical basis for each stage of approximation, Appendix 1 sets out the equations of motion relevant to each stage.

STAGE 1: LINEARIZED INVISCID

Numerical computation methods using this stage of approximation are generally termed "panel" or "vortex lattice" methods[4]. Flows over aerofoils and simple body shapes have been computed within the framework of this stage of approximation since the early 1930's. Since the 1960's it has been practical to use these methods for complete aircraft configurations. This stage of approximation could therefore

be labelled the "past" stage, and computer codes using methods could now be considered to be mature, although their improvement still continues.

Whereas the full equations of motion representing conservation of mass, momentum and energy contain altogether 60 partial-derivative terms, the steady linearized inviscid approximation truncates this to the well known potential wave equation containing only 9 terms. It is remarkable that such a seemingly crude approximation should prove so practically useful. For subsonic subcritical flow over configurations without separation, these methods provide realistic results for pressure distributions, body forces and moments including induced vortex drag, but cannot of course predict the viscous contribution to total drag.

Computer requirements for this stage of approximation are relatively modest. Calculations of complete aircraft configurations can be economically handles by computers in the IBM 360, PDP10, or CDC 6400 class (i.e. speeds in the range 0.01 to 0.1 Mflops (millions of floating point operations per second) and a memory capacity of 10^4 to 10^5 words). Even in this case the degree of resolvable geometric detail (i.e. number of panels used) is limited by computer speed, since computation time increases at between n^2 and n^3 where n is the number of panels.

STAGE II: NONLINEAR INVISCID

In this stage of approximation, only the viscous terms in the equations of motion are neglected, retaining some 27 of the 60 partial-derivative terms of the full equations.

Work in this area began in earnest with the sudden upsurge of interest in the calculation of transonic flows in about 1970. Up to that time very few computations of practical transonic flowfields had been possible and most of those which had been attempted involved hand relaxation techniques. As would be expected, the early work concentrated on simpler two-dimensional aerofoils[5,6] and bodies of revolution[7], but codes were quickly developed to treat isolated wings[5], axisymmetric bodies at angle of attack[8] and wing-body combinations[9].

The major advance over Stage I approximations is the ability to treat the nonlinear equations and hence applicability to transonic flow, including prediction of transonic wave drag. The neglect of viscous terms of course still limits these methods to flow fields without separation and precludes the estimation of skin friction drag.

This stage of approximation is still under vigorous development and could therefore be labelled the "present" stage. Methods for treating aerofoil and body of revolution flows could be considered

mature. Isolated wing and wing-body solutions are fast approaching maturity while the application to full aircraft configurations is still some years off. A significant amount of effort has been expended in attempts to combine solutions for aerofoils and isolated wings with some form of optimization system to perform the "indirect" design problem, and such attempts have met with limited success.

Being nonlinear, these methods place a much greater requirement on computer speed than does the linear approach of Stage I. Simple two-dimensional geometries are within the capabilities of the IBM360 CDC6600 class of machines, with computation times of the order of 30 minutes per case. Such computation times of course preclude the use of optimisation routines where many cases would need to be calculated and total times would quickly become impracticable. Simulation of wing and wing-body flows requires computer speeds available from CDC 7600 or Cray 1 class of machine (about 5-10Mflops). Even there the amount of memory available (up to 10^6 words) is inadequate for the three-dimensional grid densities required to achieve the desired spatial resolution. It has been estimated that to solve a wing-body flow on the PDP-10 using a reasonable (100x100x100) grid spacing, would require approximately 30 hours of computing time and this would be increased still further through having to use some form of slow storage device due to inadequate memory - clearly not a viable undertaking. It is clear that even the largest of today's super computers are barely fast enough to conduct simulations of full aircraft configurations economically, especially if the iterative process of optimization is included.

STAGE III: BOUNDARY LAYER

Prandtl's boundary layer hypothesis amounts to the statement that there is a thin layer of fluid in the neighbourhood of solid boundaries within which the forces of viscosity and inertia are of comparable magnitudes, whereas outside this layer the effects of viscosity are negligible, and the fluid behaves as if inviscid. To obtain the boundary layer equations, the Navier-Stokes equations are first subject to the Reynolds averaging process, in which the velocity is assumed to be the sum of mean and fluctuating components. Prandtl's boundary layer approximations are then included - that based on the boundary layer thickness remaining small, pressure variations normal to the surface may be ignored and viscous normal stresses will be much smaller than shear stresses and can also safely be neglected.

Unfortunately, averaging the Navier-Stokes equations eliminates some of the information they originally contained, increasing the number of unknowns above the number of variables, by substituting apparent mean (Reynolds) stresses for the actual process of transfer of momentum by the velocity fluctuations. The problem then becomes one of supplying the missing information by formulating a model to describe some or all of the six independent Reynolds stresses - the so called

closure problem. All but the very simplest closure methods require empirical information about turbulence quantities. Generally these quantities have not been experimentally measured to the accuracy needed for calculation methods, if indeed they have ever been measured at all.

Stage III approximations are usually used in combination with the inviscid approaches of either Stage I or II. The boundary layer equations are solved for the flow in the thin layer close to the body where viscosity is important, and this solution is then used to define an equivalent "inviscid" body about which the potential flow is then calculated. Interaction between the inviscid outer flow and the boundary layer is included by iterating between the two solutions until convergence is reached.

Historically this stage of approximation is older than that of Stage II, hand calculations of the boundary layer equations having been conducted since the 1930's. The computer however has had two major impacts on this stage of approximation. It has reduced the computational constraints on the choice of turbulence model, allowing the use of more sophisticated closure models such as the second order or Reynolds stress closure, improving accuracy and extending applicability when compared with the eddy-viscosity or mixing length closure models of the early hand calculations. The computer has also eased the computational burden of iterative boundary-layer/inviscid combined solutions making such solutions a practical reality.

Although boundary-layer approximations take the effects of viscosity into account, such methods still suffer from fairly severe limitations. Even if the closure problem could be satisfactorily solved (and our knowledge of turbulence will need to increase greatly before this is the case), this stage of approximation is still firmly based in the Prandtl boundary-layer hypothesis. Thus, solutions are restricted to cases where the boundary-layers remain thin, and this precludes any consideration of cases involving flow separation, large crossflows, or large pressure gradients (for example in flows including shock waves). In other words they are restricted to flows in which the viscous/inviscid interactions remain weak.

Present day boundary-layer methods can predict with fair accuracy flows over simple two-dimensional geometries while the above restrictions are satisfied. For example, reliable solutions for both the direct and inverse problems may be made for transonic aerofoil flows by a combination of Stage II and III methods[10,11] while any shock waves remain weak and the flow completely attached. The situation in three dimensions and flows over more complex geometries however has not yet reached the stage of development to be useful as predictive methods.

The computer requirements of Stage III methods vary widely. For straightforward boundary layer calculations on two-dimensional or even three-dimensional geometries, present day computer capacities are generally more than adequate. However when boundary layer methods are combined with a Stage I or II potential method in an interactive situation, several iterations of the potential flow solution are generally necessary and computation times increase accordingly. For the two-dimensional aerofoil, for example, computation times have been found to increase by a factor of between 5 and 10. Hence, although current computers are adequate for the two-dimensional case, the ability to simulate full aircraft flows with these methods would severely tax even the largest of the current supercomputers. Thus the full potential of such methods will not be realized until the next generation of computers becomes available.

STAGE IV: REYNOLDS AVERAGED NAVIER STOKES

In cases where the boundary-layer approximations break down, the methods of Stage III must be discarded and a new level of approximation found. If the same process of time averaging to the Navier Stokes equations as in Stage III, is applied but without applying the boundary-layer approximations, the so called "Reynolds Averaged" Navier-Stokes equations are obtained. In this case the basic equations are time averaged over a time interval which is long compared to turbulent eddy fluctuations yet small compared to macroscopic flow changes. The resulting equations do not neglect any terms in the full Navier-Stokes equations, but as in the Stage III approximation, the time averaging process destroys some of the information contained in the original equations, and this information must once again be modelled. The primary merits of this approximation are that it can provide realistic simulations of separated flows, of unsteady flows such as buffeting, and of total drag, including the contribution from viscous effects.

In simulating flows using Stage IV approximations, the same equations are solved throuhgout the flow field. However, although the grid spacing in the streamwise direction (along the body surface) is similar to that used for inviscid flows, the grid spacing normal to the surface is very much finer in the vicinity of the surface in order to resolve the thin viscous layer typical of high Reynolds number flows. This immediately increases the computer memory requirements over that for an inviscid solution. The memory requirements are further increased by the greater number of terms included in the equations being solved, and together these effects account for about an order of magnitude increase in memory when compared to a non-linear inviscid simulation. The combination of the greater complexity of the equations to be solved and the larger number of grid points will also drastically increase computation time, best estimates indicating an increase of almost two orders of magnitude over non-linear inviscid methods.

Clearly such requirements are beyond todays supercomputers for all but the simplest of two-dimensional problems. Routine calculations of complex three-dimensional flows will have to await the development of new computers with capabilities at least two orders of magnitude greater than today's supercomputers.

Hence, this stage of approximation is clearly a "future" one, awaiting the development of greater computer power before it can be developed to its full potential. However, perhaps more importantly, development of these codes depends on the development of more accurate turbulence models. It would appear that the major use of Stage IV codes in the future will in fact lie in improving turbulence modelling by incorporating the latest developments and running test cases, the excessive run times being practical for this type of work where they would not be justified for routine engineering calculations of such flows.

STAGE V: LARGE EDDY SIMULATIONS

The fifth, and for all practical aerodynamic purposes, final stage of approximation involves solving the full time-dependent Navier-Stokes equations in their entirety. In this stage, all the significant-sized turbulent eddies in a given flow would be computed for a sufficiently long time to yield both the time-averaged and unsteady characteristics of the flow. Fortunately the eddies that transport the principal part of the momentum and energy in a boundary layer are large, of the order of ten or more boundary layer thicknesses. They are the eddies that extract energy from the mean flow, they are highly anisotropic, and are variable from flow to flow. The small eddies on the other hand dissipate energy, are assumed to tend towards isotropy and to be nearly "universal" in character, and play little part in the transport of turbulent energy or momentum. Thus in this stage of approximation of the Navier-Stokes equations, the large eddies would be calculated, but the small subgrid scale eddies are modelled. Under these circumstances, if the grid size was small enough that the end results were insensitive to the particular subgrid scale eddy modelling employed, the results would represent computations from first principles with essentially no empiricism involved.

Such simulations would be extremely demanding on computer speed and memory capacity, at least several orders of magnitude more than Stage IV approximations. By present standards this is an extraordinarily large amount of computer time and storage, and the development of advanced computer concepts clearly paces the achievement of such simulations for practical aerodynamic configurations.

The large eddy simulation approach is however presently being used to investigate the fundamental character of turbulent motion by running simulations of very simple configurations. The aim is to

develop more general alternatives to the Reynolds stress approach to closure of the Reynolds averaged equations of Stage IV. One example[12] of such cases presently being studies is that of the flow in a simple rectangular duct at a Reynolds number (based on duct half width) of 14000. Grid size was 65x16x16. By averaging the instantaneous values of streamwise velocity and calculating the Reynolds stress, and comparing it with that of the subgrid model contribution, a test of how well the energy containing eddies are being captured can be conducted. In this case, although the shear stresses were adequately represented, the grid was not fine enough to properly represent the normal stresses. Even then, the time to compute just two periods of the large eddy structure oscillation using all the memory of a CDC 7600 was over 20 hours (about 1000 hours - 40 days and 40 nights - on the PDP-10).

The most important assumption involved in Stage V simulations is that the subgrid scale turbulence is indeed isotropic, since that is the only type of turbulence which can be successfully modelled from first principles. Although there is some evidence to support this assumption, recent research into the basic structure of turbulence has introduced some doubt that the smallest scales ever approach isotropy. If this proves to be the case, then the whole approach to large eddy simulation described above is invalid. There are other approaches, such as those which use discrete vortices to represent the turbulence (vortex dynamic simulations) but a great deal of fundamental work on the fine structure of turbulence will obviously be needed before Stage V type simulations can be used with complete confidence.

The simulations of Stage IV have been labelled as "future" methods, perhaps even "near future". Clearly Stage V approximations belong to the "much more distant future" with very little chance of them being used for routine aerodynamic calculations until well into the next century.

COMPUTATIONAL FLUID DYNAMICS IN THE FUTURE

What developments in the field of computational fluid dynamics can realistically be expected in the next 10 to 20 years? It is said that NASA aims to have "solved" the complete Navier-Stokes equations by the year 2000, and taking their dedication to the task of putting a man on the moon as a guide, it would be a brave man who categorically stated that this aim was impossible. As seen in the previous section, the major pacing item in this area is the development of computers with about 10^6 times the speed of current machines. The manner in which such machines would be developed has been the subject of much discussion in the literature. The major problem seems to be that machines of this size will be required by so few potential users as to make their development by private industry uneconomic, thus requiring some form of government intervention. The magnitude of the

gap between such huge new machines and even the largest of today's supercomputers is graphically illustrated in Figure 2. Indeed, whether the development of such machines is even physically plausible is still the subject of debate. The views of authors from outside NASA are generally much less optimistic, and tend to the conclusion that aerodynamically useful solutions of the full Navier-Stokes equations will not be available until well into the next century, if at all.

What then of developments in the earlier stages of approximation to the full equations of motion? Obviously as bigger and faster computers become available, the area in which cost effective simulations are possible will continue to expand. For Stage I and II approximations this will mean grid sizes will expand enabling better and better resolution of the details of the flow. Solutions to the "indirect" problem will become more common and "optimized" solutions will be obtained for a much wider and more complex range of configurations than is presently possible. For Stage III and IV approximations, as well as increased computer power, there is a crucial need for development of new and more accurate turbulence models to solve the closure problem. This will require much more effort to be applied to experimental investigation of the basic nature of turbulence. Otherwise turbulence models will begin to rely too heavily on trial and error adjustment of empirical constants, leading in Bradshaw's words "to the unhealthy situation of too many computers chasing too few facts". (Figure 3).

It seems likely that while the memory capacity and speed of available computers remains insufficient for a complete solution to the Reynolds Averaged equations of motion (Stage IV), a new substage will appear, between Stages III and IV. It is possible to remove some or all of the restrictions introduced at Stage III by the boundary layer assumptions (for example by allowing the pressure to vary normal to the wall) to obtain a more accurate simulation of real aerodynamic flows. If the Stage III approach of only including viscous effects in the solution of the boundary layer and then iterating between boundary layer and outer potential flow solution is retained, a significant saving in memory is possible over a solution to the full Reynolds Averaged equations throughout the flow. Although this would not solve all the probelms of the Stage III approach, it would be a useful interim measure, and indeed such schemes are now appearing in the literature[13].

To take full advantage of increases in available computer power and advances in numerical algorithms, development will be required in several peripheral areas of numerical analysis. With increased memory size, there will be a tendency for much larger grids to be employed both for existing as well as future levels of approximation. This will necessarily require the development of systems such as

automatic grid generation and input geometry packages to avoid the situation where the user is simply overwhelmed by the magnitude of the task of specifying grid geometry and boundary conditions. Much work will also be required in adapting algorithms to take full advantage of the speed of the newer parallel processing computers. To realize the full potential speed advantage of these machines, algorithms must be designed so that a very large percentage (up to 99%) of calculations are conducted in parallel mode[14]. An associated problem area is that of designing new high level programming languages to take advantage of parallel computation. Traditionally, most computational fluid dynamics problems have been programmed using FORTRAN or similar high level languages - languages designed essentially for serial computation. Hence, before full advantage can be taken of even presently available supercomputers, the computational fluid dynamicist must rely on developments in areas not directly related to fluid dynamics.

There is no doubt that in the next one or two decades developments in computational fluid dynamics will revolutionize the areas of aerodynamics design and analysis. Whereas with past and present potential flow calculations the analyst/designer himself had to input many crucial inputs (such as transition point, shockwave and separation locations, viscous/inviscid interactions) future numerical simulations will allow treatment of many such problems automatically. There will also be a much greater emphasis on the simulation of much more complex three-dimensional configurations, with correct representation of viscous effects.

IMPACT OF COMPUTATIONAL FLUID DYNAMICS ON A.R.L.

The use of computational fluid dynamics at ARL has, up to the present time, been restricted to the first three stages of approximation to the full Navier-Stokes equations, and consideration has been limited almost entirely to two-dimensional configurations. The major reason for these restrictions has been the limited computer power available in the DECSystem10. Experience of the author with running coupled non-linear inviscid/boundary-layer solutions for the transonic flow over two-dimensional aerofoils has indicated that even for this relatively simple case, the run times required (in this case several hours) quickly make the process extremely frustrating, if not totally impracticable. From the run times of examples of the later stages of approximation quoted earlier in this paper, it is clear that the computing power at present available at ARL is completely inadequate for any more ambitious computations.

The impact of computational fluid dynamics on ARL will rely on the answers to two inter-related questions: In what areas and to what extent will computational fluid dynamics be used at ARL? and, What type of computer will be required to conduct these simulations?

The answers to these two questions are obviously dependent upon each other since the type of work undertaken will largely determine the size of computer required. Equally however, the type and capacity of the available computer will control the range of simulations which are possible.

The answer to the first question is the more difficult to predict, since it is intimately connected with the type of project to be undertaken in the future at ARL. Certainly it appears unlikely that there will be much ab initio aircraft design in Australia, and as stated earlier it would be in this area that the potential benefits of computational fluid dynamics would be most obvious. Most probably ARL's role will remain much as it is today, in which case it is possible to indicate two main areas in which computational fluid dynamics will be involved:

(i) To complement and enhance wind tunnel investigations.

Apart from their use as research tools, the use of wind tunnels at ARL, has generally been confined to two major areas. Firstly, for the measurement of the aerodynamic characteristics of newly developed aircraft and missiles e.g. Jindivik, Ikara, Nomad, where the wind tunnel tests are used as verification of the design procedure. Computational fluid dynamics is expected to have no direct impact on this area, since the development of design methods to a stage where no such verification is required, even with the assistance of greatly improved computer simulations, still seems a long way off. Secondly, wind tunnels have been used to investigate problems that arise from the operation of existing military aircraft, such as the carriage of non-standard stores or operation outside the normal flight envelope. Typical problems in this area involve strong viscous interactions with complex three-dimensional separations and often include the presence of shock waves. The development of computational fluid dynamics methods which can accurately model these effects will require Stage V type simulations and as stated earlier, such simulations will probably not be available until at least early in the next century.

Thus developments in computational fluid dynamics are not expected to have a significant direct impact on the type of wind tunnel testing carried out at ARL in the foreseeable future, and will in no way lessen the need for new or upgraded wind tunnel facilities. Rather it would be expected that there will be a complementary use of the computer and the wind tunnel whereby the strengths of one are used to supplement the other. Areas of potential use for computational methods could include the confirmation or support of otherwise uncertain conclusions drawn solely from experiment, as an aid in the extrapolation of wind tunnel test results to full scale, and to investigate and if possible exclude unwanted effects due to wall interference and other extraneous influences.

(ii) As a research tool.

This area is readily broken into two. Firstly where computational methods are used as an aid to the understanding of some physical process, for example, the use of Large Eddy simulations as a tool to assist in the understanding of the structure of turbulence. Secondly, there is the development of computational methods themselves. Traditionally ARL has conducted fairly basic work in this area (e.g. ref.15), but in the future it would seem likely that this will not continue. Due to a combination of the small effort available, and the increasing complexity of codes (in a numerical analysis sense), it seems more likely that ARL effort in this area will be directed towards improvement and verification of existing codes. In fact ARL's size and facilities make it uniquely fitted to fill such a role, especially that of improvement of the physical bases of codes - an area which is often sadly neglected by overseas institutions in their haste to produce working codes.

Turning to the question of required computer power, as stated above, the driving force here is the complexity of the simulation to be undertaken. It is safe to assume that for the uses described in (1) above only "mature" codes will be used. This is partly due to the need for a high degree of confidence in a particular code before being used for such tasks, but more practically, since ARL will rely almost totally on overseas sources for acquiring new codes, it is only "mature" codes that can generally be obtained. For use as research tools, the same degree of confidence is not required but the problem of availability must still be overcome. Generally it seems that as far as "production" codes are concerned, use at ARL will lag use by the U.S. aircraft companies by about five years.

What then does this mean for the type of computer required for computational fluid dynamics at ARL in the next ten to fifteen years? A good idea of the codes being used by US aircraft companies in 1977 can be obtained from their contributions to Reference 16. From these, it is clear that three-dimensional versions of Stage II simulations were being used on a production basis and that two-dimensional Stage IV simulations were at a developmental stage. Both these tasks exceeded the capacity of the largest general purpose computer available to them - the CDC 7600. From Reference 16 it is also clear that the aircraft companies supported the development of the NASF* and indicated their willingness to purchase such a machine (200 times faster than the CDC 7600) when it became economically justified. (The NASF is expected to be operational in about 1985).

*Numerical Aerodynamic Simulation Facility

Thus it can be expected that the codes that will be available to ARL in fifteen years time will require machines of the capacity of NASF to be cost effective. Current proposals for updating the ARL DECSystem10 consider speed increases of less than one order of magnitude - compared with a four order of magnitude increase required to match the NASF. Clerarly a machine of the capacity of NASF is a very expensive piece of equipment (of the same order as a new wind tunnel) and could not be justified on the basis of the needs of ARL alone. Indeed there seems little doubt that the purchase of a machine of that size would need to be based on national needs, with computational fluid dynamics forming quite a small part of its projected work load.

The most important objective for ARL will therefore be to ensure that consideration is given to the purchase of such a machine, and to advocate the continual updating of the largest machines currently available in this country. Equally importantly, ARL must be guaranteed access to such large computers. Access must be both physically simple (e.g. the provision of on-line terminals at ARL) and the charges for use of the machine must be realistic. There is absolutely no point in having access to a large computer if permission must be sought from Canberra before running a job because of the high costs involved.

CONCLUSIONS

Computational fluid dynamics is a relatively new but rapidly expanding scientific discipline. The last two decades have seen it develop from being limited to simple two-dimensional solutions of the inviscid linear approximations of the full governing equations, to the situation today, where full three-dimensional aircraft configurations may be considered, including the effects of viscosity and of shock waves in the transonic speed range. Prospects for further great developments in the future appear good, although whether solutions to the full Navier-Stokes equations in their complete form will ever be possible still remains in doubt. The newer levels of approximation require quite extraordinary computer capacity; present developments have already outstripped the capacity of even the largest supercomputers currently available. The development of faster and larger computers will be one of the major pacing items in the future advances in this area.

This extraordinary development has led to a tremendous increase in the usefulness of computational fluid dynamics. Numerical methods have now reached a stage where they can complement and extend the results available from wind tunnel testing. However, there seems little likelihood of them reaching the stage of making wind tunnels obselete for a considerable time, if at all.

The use of computational methods at ARL is presently severely restricted by the computer power currently available. To take full advantage of the rapidly increasing power of computational fluid dynamics, a computer of much greater capacity than that likely to be available at ARL in the next ten to fifteen years will obviously be necessary. To justify the purchase of a computer to suit the needs of ARL in this area will require th combination of the requirements of potential users throughout the country, both in this area and in other areas requiring the provision of such a machine (e.g. meteorologists). However, it is very much in ARL's interest to encourage the purchase of such a machine, and if successful to ensure adequate access.

REFERENCES

1. Chapman, D.R.

   Dryden Lecture: Computational Aerodynamics Development and Outlook.
   AIAA J. 17,2, 1293-1313, 1979

2. Sloof, J.W.

   Wind Tunnel Tests and Aerodynamic Computations: Thoughts on their Use in Aerodynamic Design.
   AGARD CP210. Numerical Methods and Windtunnel Testing, 1976.

3. Bradshaw, P.

   The Understanding and Prediction of Turbulent Flow.
   Aero. J., 76, 403-416, 1972.

4. Woodward, F.A.

   Analysis and Design of Wing-Body Combinations at Subsonic and Supersonic Speeds.
   J. Aircraft, 5, 528-534, 1968.

5. Jameson, A.

   Iterative Solutions of Transonic Flows over Aerofoils and Wings, Including Flows at Mach 1.
   Comm. Pure and Appl. Math., 27, 283-309, 1974.

6. Garabedian, P.R., Korn, D.G.

   Analysis of Transonic Aerofoils.
   Comm. Pure and Appl. Math., 24, 841-851, 1972.

7. South, J.C., Jameson, A.

   Relaxation Solutions for Inviscid Axisymmetric Transonic Flow over Blunt or Pointed Bodies.
   A.R.C. 34-726, 1973.

8. Reyhner, T.A.

   Transonic Potential Flow around Axisymmetric Inlets and Bodies at Angle of Attack.
   AIAAJ, 15, 1299-1306, 1977.

9. Caughey, D.A., Jameson, A.

   Numerical Calculation of Transonic Potential Flow About Wing-Body Combinations.
   AIAAJ., 17, 175- , 1979.

REFERENCES CONTINUED

10. Bavitz, R.

    An Analysis Method for Two-Dimensional Transonic Viscous Flow.
    NASA TN-D-7718, 1974.

11. Collyer, M.R.

    An Extension to the Method of Garabedian and Korn for the Calculation of Transonic Flow Past an Aerofoil to Include the Effects of a Boundary Layer and Wake.
    RAE-TR-77104, 1977.

12. Moin, P., Reynolds, W.C., and Ferziger, J.H.

    Large Eddy Simulation of a Turbulent Mixing Layer,
    Standford Univ. Rept. TF-11. 1978.

13. Mahgoub, H.E.H., Bradshaw, P.

    Calculation of Turbulent-Inviscid Interactions with Large Normal Pressure Gradients.
    A.R.C.37-572, 1977.

14. Redhed, D.D., Chen, A.W., Hotory, S.C.

    New Approach to the 3-D Transonic Flow Analysis Using the STAR-100 Computer.
    AIAAJ., 17, 98-100, 1979.

15. Georgeff, M.P.

    Application of Finite Difference Methods to Wholly Subsonic Axisymmetric and Plane Flows.
    ARL - Aerodynamics Note 326, 1970.

16.

    Future Computer Requirements for Computational Aerodynamics - A Workshop.
    NASA-CP-2032, 1978.

Wind tunnel	Computational fluid dynamics
1. Complete physics (full equations of motion)	1. Partial physics (approximate equations of motion)
2. Incorrect geometrical environment (walls, stings, etc.)	2. Correct geometrical environment is possible
3. Model changes time consuming and expensive	3. Model geometry relatively easily changed
4. Easy to change flow conditions	4. Changes in flow conditions may require different models
5. Flow about a given body only	5. Type of boundary condition may by changed
6. Limited accessibility	6. High accessibility

FIG 1 COMPARISON OF ATTRIBUTES OF WIND TUNNELS AND COMPUTATIONAL FLUID DYNAMICS

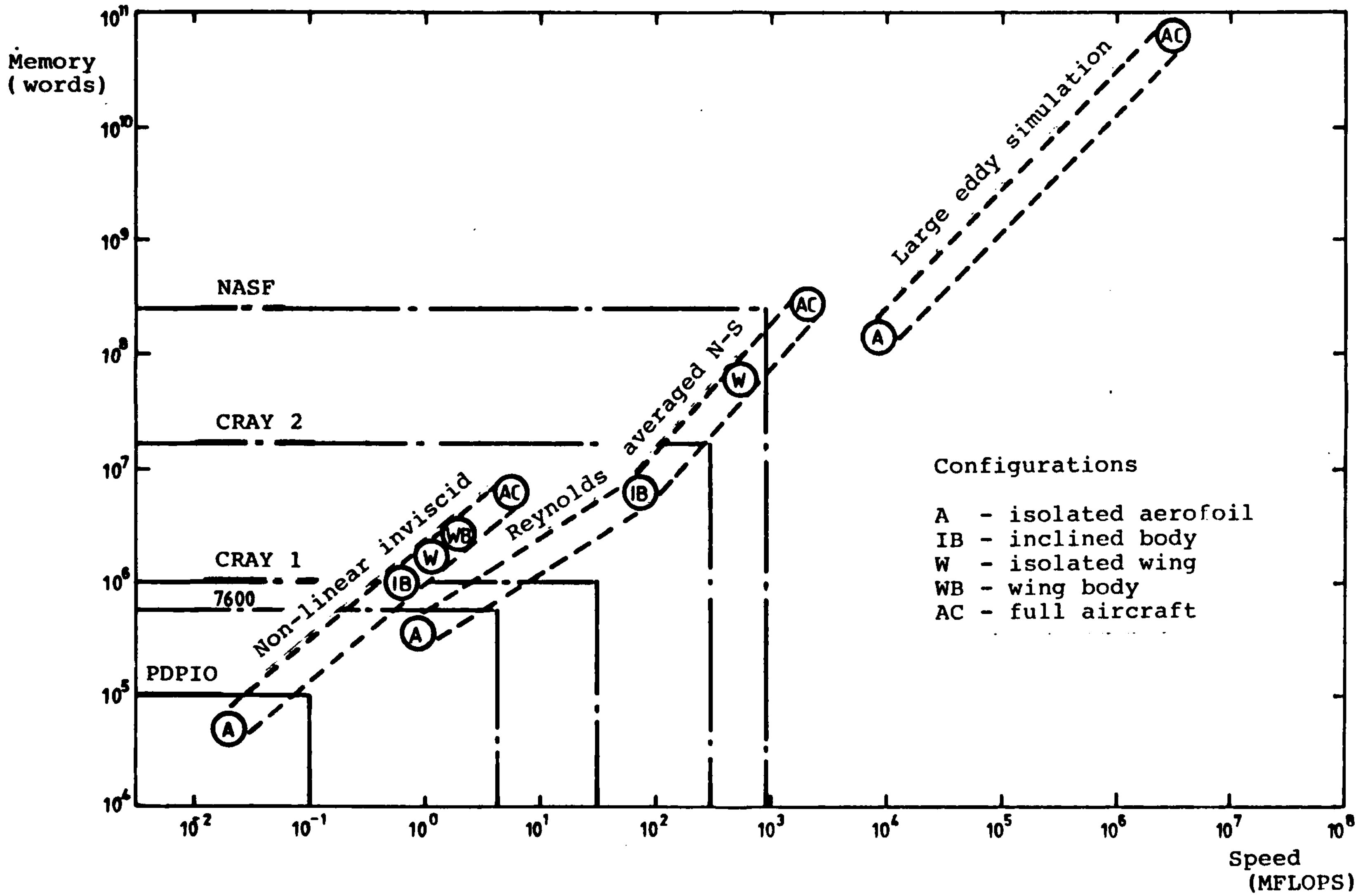


FIGURE 2. COMPUTER REQUIREMENTS

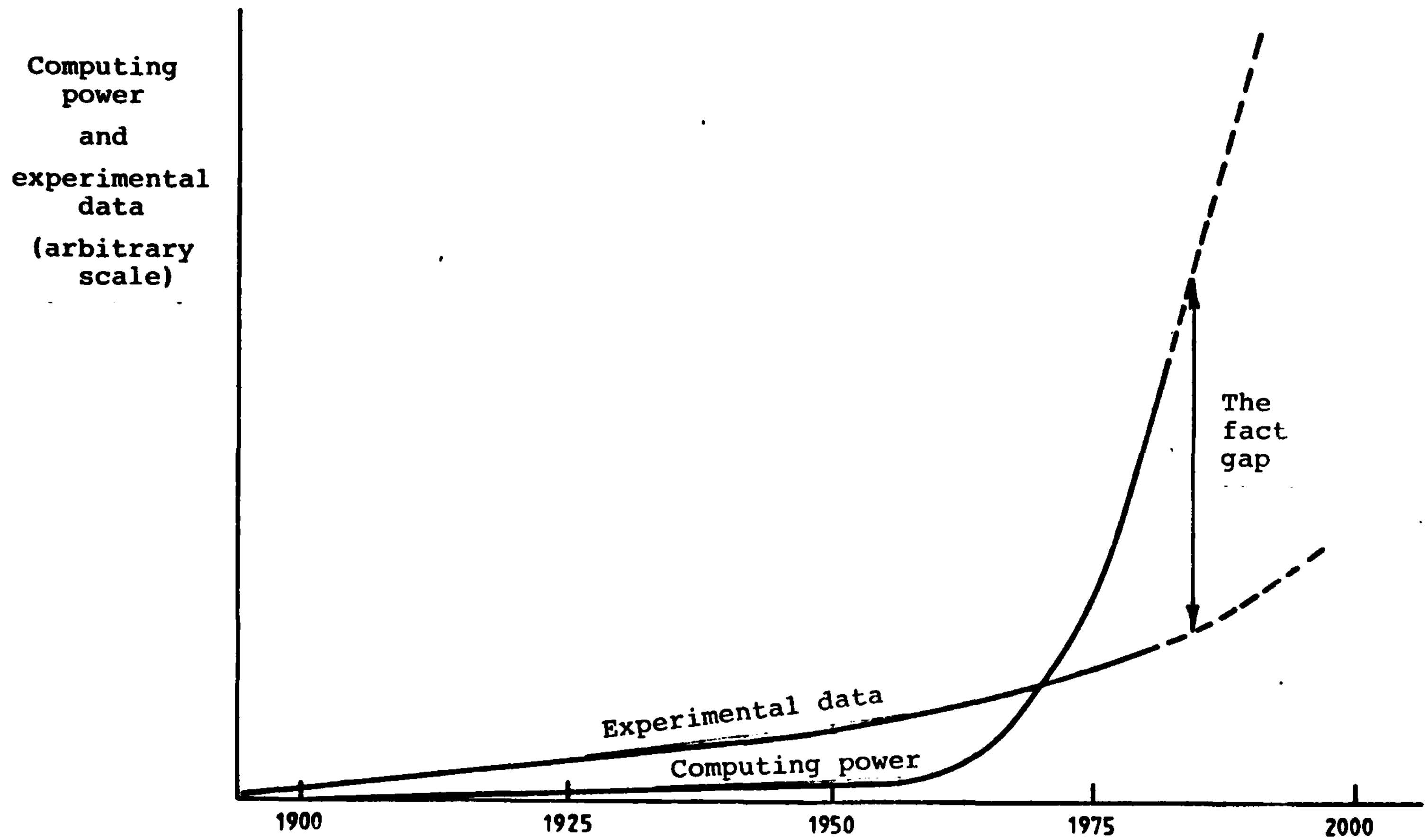


FIG.3 THE FACT GAP

APPENDIX: THE NAVIER STOKES EQUATIONS AND APPROXIMATIONS TO THEM

The equations governing the motion of a compressible viscous fluid have been well known for many years. For a general three-dimensional motion, the velocity field is specified by the velocity victor V with components in orthogonal coordinates u, v, and w, by the pressure p and the density ρ. For these five unknown quantities there exist five equations: the continuity equation (conservation of mass) the three equations of motion (conservation of momentum) and the thermodynamic equation of state. (Note that if the equation of state- which relates pressure and density - also contains temperature as an additional variable, a further equation is supplied by the principle of conservation of energy).

In its most general form, the continuity equation may be written

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad \text{—— (1)}$$

If the fluid may be considered to be continuum (which is true for air at normal atmospheric comditions close to sea level), is isotropic and Newtonian, then the form of the momentum equations which may be derived are usually referred to as the Navier-Stokes equations even though the original derivations applied only to incompressible flow. These equations may be written for the x-component as

$$\begin{aligned}
&\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \\
&\rho X - \frac{\partial p}{\partial x} + \mu\left\{\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right\} \\
&+ \frac{1}{3}\mu\left\{\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial x \partial y} + \frac{\partial^2 w}{\partial x \partial z}\right\} \\
&+ \frac{\partial \mu}{\partial x}\left\{\frac{\partial u}{\partial x} + \frac{\partial u}{\partial x}\right\} + \frac{\partial \mu}{\partial y}\left\{\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right\} + \frac{\partial \mu}{\partial z}\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\} \\
&- \frac{2}{3}\frac{\partial \mu}{\partial x}\left\{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right\} \qquad \text{—— (2)}
\end{aligned}$$

with corresponding equations for the y- and z-components. Here X is the x-component of an external body force per unit mass and μ is viscosity.

For an incompressible fluid (i.e. ρ = cost.) these equations can be greatly simplified, since in this case temperature variations are generally small and viscosity may also be taken to be constant.

The continuity equation then becomes

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad \text{—— (3)}$$

and the x-component momentum equation becomes

$$\rho \left\{ \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \right\} =$$

$$\rho X - \frac{\partial p}{\partial x} + \mu \left\{ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right\} \qquad \text{—— (4)}$$

Note that in this case the equation of state as well as the energy equation become superfluous as far as the calculation of the flow field is concerned.

Returning to the complete equations (1) & (2), the Reynolds averaged Navier Stokes equations are obtained by expressing the velocity components, pressure and density as sums of mean and fluctuating quantities:

$$u = \bar{u} + u'$$

etc.

where

$$\bar{u} = \frac{1}{T}\int_0^T u \, dt$$

and T is a period of time which is large when compared to the characteristic time scale of the turbulent motions.

Substituting into equation (1) produces

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\bar{u}}{\partial x} + \frac{\partial \bar{\rho}\bar{v}}{\partial y} + \frac{\partial \bar{\rho}\bar{w}}{\partial z}$$

$$+ \frac{\partial}{\partial x}\overline{\rho' u'} + \frac{\partial}{\partial y}\overline{\rho' v'} + \frac{\partial}{\partial z}\overline{\rho' w'} = 0 \qquad \text{—— (5)}$$

and equation (2) becomes

$$\bar{\rho}\left\{\frac{\partial\bar{u}}{\partial t}+\bar{u}\frac{\partial\bar{u}}{\partial x}+\bar{v}\frac{\partial\bar{u}}{\partial y}+\bar{w}\frac{\partial\bar{u}}{\partial z}\right\}+\frac{\partial}{\partial t}\overline{\rho'u'} =$$

$$\bar{\rho}X-\frac{\partial\bar{p}}{\partial x}+\mu\left\{\frac{\partial^2\bar{u}}{\partial x^2}+\frac{\partial^2\bar{u}}{\partial y^2}+\frac{\partial^2\bar{u}}{\partial z^2}\right\}$$

$$+\frac{1}{3}\mu\left\{\frac{\partial^2\bar{u}}{\partial x^2}+\frac{\partial^2\bar{v}}{\partial x\partial y}+\frac{\partial^2\bar{w}}{\partial x\partial z}\right\}+\frac{\partial\mu}{\partial x}\left\{\frac{\partial\bar{u}}{\partial x}+\frac{\partial\bar{u}}{\partial x}\right\}$$

$$+\frac{\partial\mu}{\partial y}\left\{\frac{\partial\bar{u}}{\partial y}+\frac{\partial\bar{v}}{\partial x}\right\}+\frac{\partial\mu}{\partial z}\left\{\frac{\partial\bar{u}}{\partial z}+\frac{\partial\bar{w}}{\partial x}\right\}-\frac{2}{3}\frac{\partial\mu}{\partial x}\left\{\frac{\partial\bar{u}}{\partial x}+\frac{\partial\bar{v}}{\partial y}+\frac{\partial\bar{w}}{\partial z}\right\}$$

$$-\left\{\frac{\partial}{\partial x}\bar{\rho}\,\overline{u'^2}+\frac{\partial}{\partial y}\bar{\rho}\,\overline{u'v'}+\frac{\partial}{\partial z}\bar{\rho}\,\overline{u'w'}\right.$$

$$+\frac{\partial}{\partial x}\overline{\rho'u'}\,\bar{u}+\frac{\partial}{\partial y}\overline{\rho'v'}\,\bar{u}+\frac{\partial}{\partial z}\overline{\rho'w'}\,\bar{u}+\frac{\partial}{\partial x}\overline{\rho'w'}\,\bar{u}$$

$$\left.+\frac{\partial}{\partial y}\overline{\rho'u'}\,\bar{v}+\frac{\partial}{\partial z}\overline{\rho'w'}\,\bar{w}+\frac{\partial}{\partial x}\overline{\rho'u'^2}+\frac{\partial}{\partial y}\overline{\rho'u'v'}+\frac{\partial}{\partial z}\overline{\rho'u'w'}\right\} \quad \text{—— (6)}$$

Once again, if the flow is incompressible, the equations (5) and (6) can be greatly simplified since in this case $\rho'/\rho << 1$, and the equations become:

$$\frac{\partial\bar{u}}{\partial x}+\frac{\partial\bar{v}}{\partial y}+\frac{\partial\bar{w}}{\partial z} = 0 \quad \text{—— (7)}$$

and

$$\bar{\rho}\left\{\frac{\partial\bar{u}}{\partial t}+\bar{u}\frac{\partial\bar{u}}{\partial x}+\bar{v}\frac{\partial\bar{u}}{\partial y}+\bar{w}\frac{\partial\bar{u}}{\partial z}\right\} =$$

$$\bar{\rho}X-\frac{\partial\bar{p}}{\partial x}+\mu\left\{\frac{\partial^2\bar{u}}{\partial x^2}+\frac{\partial^2\bar{u}}{\partial y^2}+\frac{\partial^2\bar{u}}{\partial z^2}\right\}$$

$$-\bar{\rho}\left\{\frac{\partial\overline{u'^2}}{\partial x}+\frac{\partial\overline{u'v'}}{\partial y}+\frac{\partial\overline{u'w'}}{\partial z}\right\} \quad \text{—— (8)}$$

In both equations (6) and (8) it may be seen that the turbulence (fluctuating) quantities have introduced an extra term on the right hand side when compared to equations (2) and (4) respectively. This term may be regarded as an apparent stress due to the turbulent fluctuations which is acting on the mean flow - the Reynolds stress. It is also apparent that the splitting up of the unsteady velocities has introduced (in the incompressible case) three extra unknowns, u', v', and w'. Although an extra equation

$$\frac{\partial u'}{\partial x}+\frac{\partial v'}{\partial y}+\frac{\partial w'}{\partial z} = 0 \quad \text{—— (9)}$$

may be derived, the number of unknowns still exceeds the number of equations and thus gives rise to the "closure problem".

The next stage of approximation to the full equations is the boundary layer equations.

The boundary layer assumptions may be stated as:-
- the boundary layer thickness everywhere remains small when compared to distances parallel to the boundary.
- rates of change normal to the boundary of any flow quantity are in general much larger than those in the streamwise direction.

For the case of a two-dimensional motion, the equation of continuity becomes

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\bar{u}}{\partial x} + \frac{\partial \bar{\rho}\bar{v}}{\partial y} + \frac{\partial \overline{\rho' v'}}{\partial y} = 0 \qquad \text{—— (10)}$$

and the momentum equation becomes

$$\bar{\rho}\left\{\frac{\partial \bar{u}}{\partial t} + \bar{u}\frac{\partial \bar{u}}{\partial x} + \bar{v}\frac{\partial \bar{u}}{\partial y}\right\} + \frac{\partial \overline{\rho' u'}}{\partial t} =$$

$$-\frac{\partial \bar{p}}{\partial x} + \mu\frac{\partial^2 \bar{u}}{\partial y^2} - \frac{\partial}{\partial y}\,\bar{\rho}\,\overline{u'v'} - \overline{\rho' v'}\,\frac{\partial \bar{u}}{\partial y} \qquad \text{—— (11)}$$

As for the Reynolds averaged equations, the number of unknowns exceeds the number of equations and empirical expressions are required to enable a solution to be obtained.

If the flow can be considered inviscid, for example in areas external to the boundary layer, the momentum equations can be greatly simplified. Setting μ=0 in the Navier Stokes equations (2) gives for the x-component of momentum

$$\rho\left\{\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right\} = \rho X - \frac{\partial p}{\partial x} \qquad \text{—— (12)}$$

with the continuity equation (1) remaining unchanged.

For calculation of transonic flows, this equation is often recast in terms of a perturbation velocity. After some algebraeic manipulation, equation (12) becomes

$$(1 - M_{oo}^2)\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = M_{oo}^2\,(\gamma-1)\,\frac{u}{U}\,\frac{\partial u}{\partial x} \qquad \text{—— (13)}$$

where M_{∞} is the free stream Mach number. Note that this equation retains all the non-linear terms - those on the right hand side - and when these are ignored the inviscid linearized equations of Stage I approximations are obtained, thus

$$(1 - M_{\infty}^2) \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad \text{—— (14)}$$

DISTRIBUTION

AUSTRALIA

Department of Defence

Central Office

Chief Defence Scientist)
Deputy Chief Defence Scientist) (1 copy)
Superintendent, Science and Technology Programmes)
Controller, Projects and Analytical Studies)
Aust. Defence Scientific and Technical Rep. (U.K.)
(Doc Data sheet only)
Counsellor, Defence Science (U.S.A.) (Doc Data sheet only)
Defence Central Library
Document Exchange Centre, D.I.S.B. (17 copies)
Joint Intelligence Organisation
Director General - Army Development (NSO) (4 copies)

Aeronautical Research Laboratories

Chief Superintendent
Library
Superintendent - Aerodynamics
Divisional File - Aerodynamics
Author: B.D. Fairlie
Transonic Wind Tunnel (10 copies)

Materials Research Laboratories

Chief Superintendent/Library

Defence Research Centre

Library

SPARES (10 copies)

TOTAL 51

DOCUMENT CONTROL DATA

1.a.AR No	1.b.Establishment No	2.Document Date	3.Task No
AR-002-342	ARL-AERO-TECH-MEMO-336	March, 1982	R6 DST 79/107

4.Title
A REVIEW OF COMPUTATIONAL FLUID DYNAMICS AND ITS IMPACT ON A.R.L.

5.Security
a.document
UNCLASSIFIED
b.title c.abstract
U U

6.No Pages
16

7.No Refs
16

8.Author
B.D. FAIRLIE

9.Downgrading Instructions
—

10.Corporate Author and Address
Aeronautical Research Laboratories

11.Authority (as appropriate)
a.Sponsor b.Security c.Downgrading
d.Approved
—

12.Secondary Distribution (of this document)

Approved for public release.

Overseas enquirers outside stated limitations should be referred through ASDIS, Defence Information Services Branch, Department of Defence, Campbell Park, CANBERRA ACT 2601.

13.a. This document may be ANNOUNCED in catalogues and awareness services available to: No limitations.

13.b. Citation for other purposes (ie casual announcement) may be unrestricted.

14.Descriptors
Computational fluid dynamics
Fluid dynamics
Computerized simulation

15.COSATI Group
0902

16.Abstract

This memorandum reviews the major developments in computational fluid dynamics. The present and future capabilities of numerical simulations are assessed to determine the demands that they will place on available computer power. The impact that these developments will have on the use of computational fluid dynamics at ARL is considered together with their implications for the provision of new computers at ARL.

17.Imprint
Aeronautical Research Laboratories, Melbourne

18.Document Series and Number
Aerodynamics Technical Memorandum 336

19.Cost Code
547710

20.Type of Report and Period Covered

21.Computer Programs Used
—

22.Establishment File Ref(s)
—

DATE

ILME

-8